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Foreword

ISO (the International Organization for Standardization) and IEC (the International Electrotechnical Commission) form a specialized system for worldwide standardization. National bodies that are members of ISO or IEC participate in the development of International Standards through technical committees established by the respective organization to deal with particular fields of technical activity. ISO and IEC technical committees collaborate in fields of mutual interest. Other international organizations, governmental and non-governmental, in liaison with ISO and IEC, also take part in the work. See http://www.iso.org for information on ISO and http://www.iec.ch for information on IEC.

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This draft was prepared by Joint Technical Committee ISO/IEC JTC 1, Information technology, Subcommittee 24, Computer graphics, image processing and environmental representation, in collaboration with the Web3D Consortium (http://www.web3d.org).

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Introduction

This draft document, Information technology — Computer graphics, image processing and environmental representation — CAD-to-X3D Conversion for Product Structure, Geometry Representation and Metadata is a technical report describing how CAD data can be converted into X3D representation for lightweight 3D visualization where two existing standards, ISO 10303 STEP and ISO/IEC 19775 Extensible 3D (X3D), are related.

The lightweight visualization data constructed from 3D CAD product data can be used for various applications such as design review, virtual operation and management simulation, 3D job order and manual, 3D part list and catalogue, 3D printing, and so on. X3D, the international standard for interactive 3D graphics in the web environment, is useful for information management and service in a distributed cooperation environment. Therefore, applying X3D to the lightweight visualization of CAD data can make it easy to share 3D product data in a distributed environment, and guarantee long lifetime with its high interoperability among heterogeneous information management systems.

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Information technology — Computer graphics, image processing and environmental representation — CAD-to-X3D Conversion for Product Structure, Geometry Representation and Metadata

1. Scope

This document reviews how STEP models can be adapted through CAD-to-X3D conversion for visualization that includes Product Structure (PS), Geometry, and Product Manufacturing Information (PMI). Metadata conventions to preserve pedigree and provenance are similarly important to support a wide variety of future Web-based applications.

Exemplars demonstrate the practical application of these technologies by applying conversion techniques to existing models using open-source software applications. The report will conclude with opportunities of shared interest for mutually beneficial work by SC24 and other standards committees.

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2. Normative references

The following referenced documents are indispensable for the application of this document. For dated references, only the edition cited applies. For undated references, the latest edition of the referenced document (including any amendments) applies.

ISO/IEC 19775-1:2013, *Information technology — Computer graphics and image processing — Extensible 3D (X3D)*

ISO/IEC 14772-1:1997, *Information technology -- Computer graphics and image processing — The Virtual Reality Modeling Language (VRML) -- Part 1: Functional specification and UTF-8 encoding*

ISO 10303-1:1994, *Industrial automation systems and integration — Product data representation and exchange — Part1: Overview and fundamental principles*

ISO 10303-21:1994, *Industrial automation systems and integration — Product data representation and exchange — Part21: Clear text encoding of the exchange structure*

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3. Terms and definitions (to be modified according to the standard format)

3.1. Terms and definitions

For the purposes of this Technical Report, the following definitions apply.

3.1.1. Part

3.1.2. Assembly

3.1.2. Sub-assembly

3.1.2. Component

3.1.2. Node

3.1.2. Metadata

3.1.3. STEP

3.1.4. CSG

3.1.5. B-REP

3.1.6. NURBS

3.1.7. Product Structure

3.1.8. Web3D

3.2 Symbols (and abbreviated terms)

For the purposes of this document, the following expansion of acronyms and abbreviations apply:

3.2.1. CAD : Computer Aided Design

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3.2.2. PS : Product Structure

3.2.3. PMI : Product Manufacturing Information

3.2.4. AM : Additive Manufacturing

3.2.5. X3D : Extensible 3D

….. more on 3D Printing and 3D Scanning . ……………..

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4. CAD-to-X3D conversion

4.1 Product Structure

4.1.1 PS information of a CAD assembly

A CAD assembly has an assembly information, or product structure (PS) which is a tree information representing the parent-child relationship between those components in the assembly. A component of an assembly can be an assembly, a sub-assembly, or a part. Even though a component may correspond to a node in a tree data structure, the term ‘node’ will not be used here to mean a component in an assembly to avoid being confused with the term ‘node’ in the X3D specification. Therefore, ‘node’ in this document will only be used as an item of the X3D specification.

For a better understanding of the conversion of PS from STEP to X3D, a Hub Assembly model as shown in Figure will be used as an example for CAD assembly data. It is modeled with CATIA CAD system, and composed of six components each of which corresponds to one file whether an assembly (sub-assembly) type format or a part type format. Hub_Assembly and sleeve_sub_assembly are respectively a root assembly and a sub-assembly components, and disc_with_holes, cap, gasket, and bushing are part components which actually have their own geometry, or shape information. A root assembly component is at the top in PS while part components are always leaves. Sub-assemblies are components between the root and the leaves. If not specifically mentioned as root or sub-assembly, assembly means both root assembly component and sub-assembly component together.

There exists transform information between every parent component and its children which is noted as ‘T’ in Figure. The transform is relative for a child component to its parent. In case of the CATIA Hub Assembly, each component is saved as a single file whether an assembly or a part, which means the relationship between component files is the same as that in PS, and a parent component has the information of external references for its children including the relative transform information for them.

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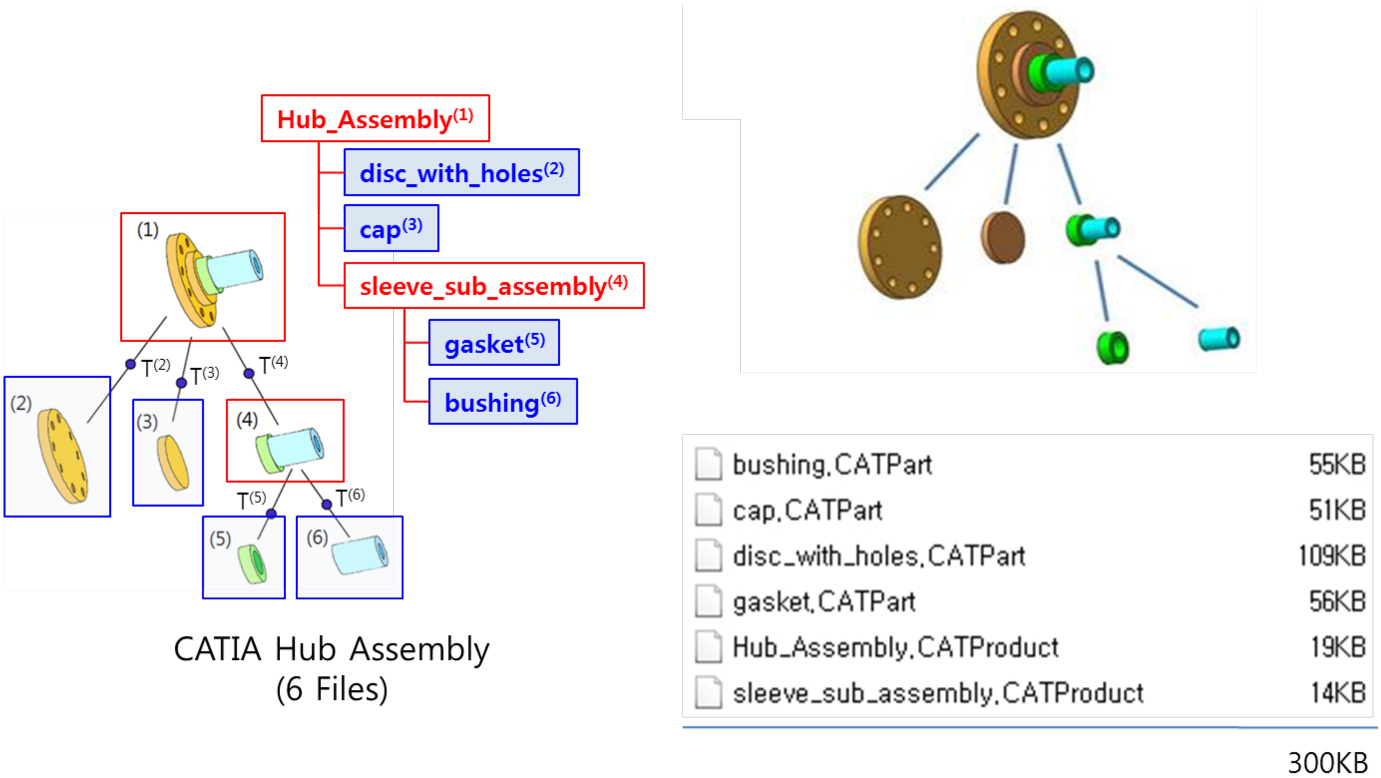

Figure: CATIA Hub Assembly model

Some legacy CAD systems provide functions for converting CAD data to VRML/X3D format such as ‘save as’ or ‘export’. The problem of conversion using these functions of the legacy CAD systems is that PS information of a CAD assembly data disappears when saved as VRML/X3D. Figure below shows that after exporting the Hub Assembly into VRML/X3D with ‘save as’ function in CATIA system, the PS information disappears and all the leaf components become children of the root component without any sub-assembly. In addition, all the data is usually saved as one single VRML/X3D file. If the PS of a CAD assembly is simple and its geometry data is not heavy there is no difficulty in handling the data. However, in case a CAD assembly’s PS is complicated and its geometry is heavy, only a part’s design change of the CAD data can cause the whole data to be converted again inefficiently.

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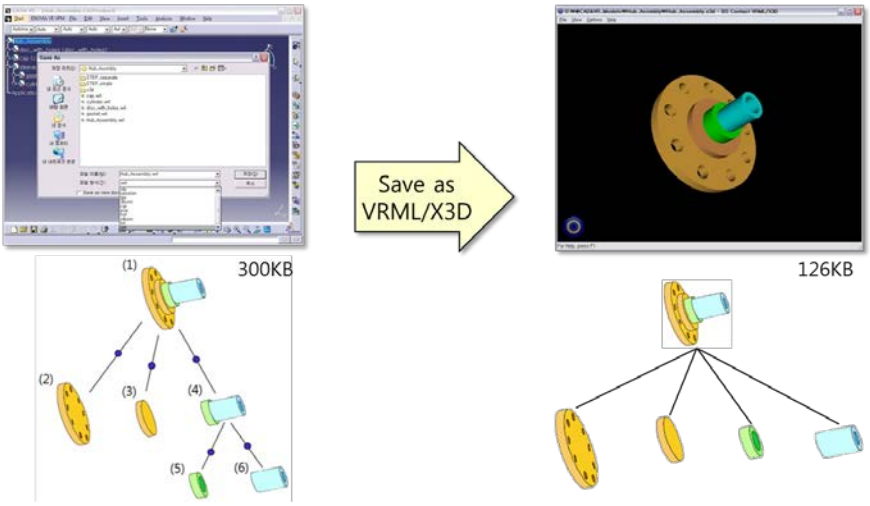


Figure: Save as VRML/X3D of CATIA Hub Assembly

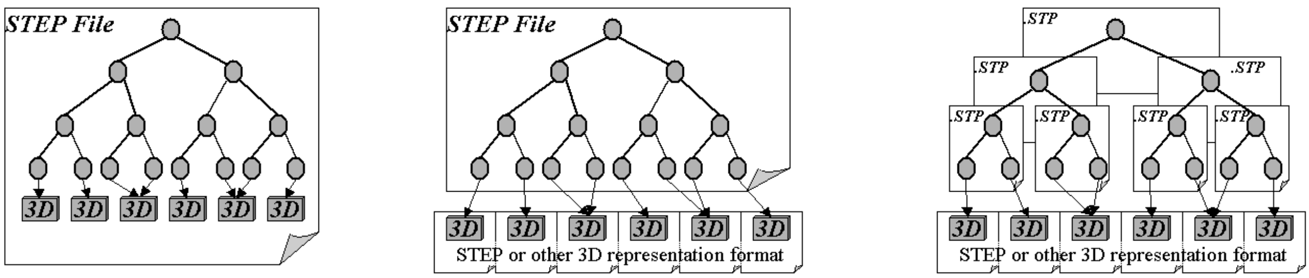
4.1.2 Representation of PS in STEP

Some more explanation ~ [2]

a) assembly and part geometries in the same file

b) an assembly file with external reference to geometry files (external reference)

c) an assembly file with externally referenced sub-assemblies and geometry files (nested external reference)



Assembly and part geometries in the same file

Assembly with external reference to 3D geometry file (in STEP or other format)

Nested assembly

Figure: STEP AP 242 functionalities for 3D CAD interoperability : Assembly [2]

In case of assembly and part geometries in the same file, STEP-based approach for extraction of PS information [6] can be applied for the extraction of PS information.

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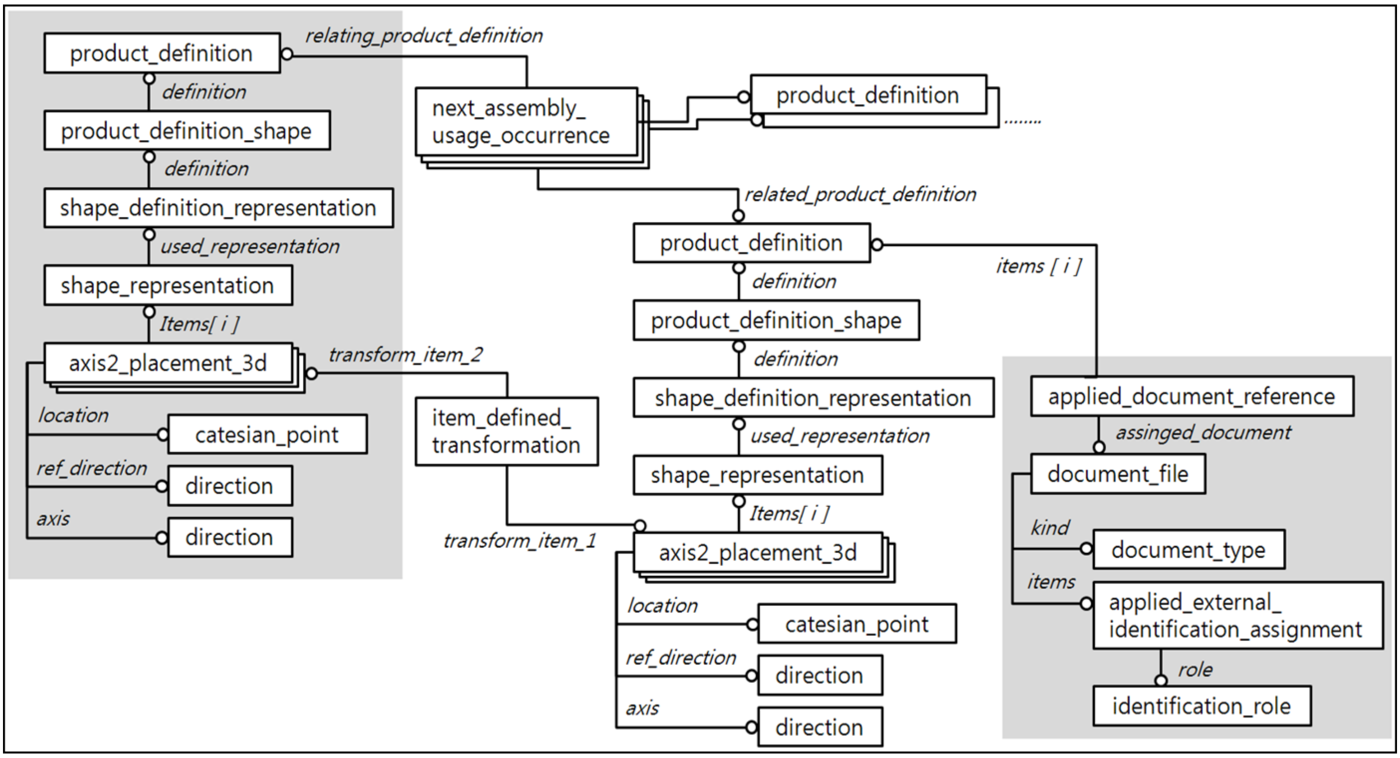

Figure: STEP AP203ed2 instance diagram for PS [6]

In case of the nested external reference or the nested external references, CAx- and PDM-IF recommended practices for external references [1] can be used for the extraction of PS information.

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Figure: Instance diagram for external references [1]

Some more details on what kinds of data extracted for X3D and mapping information should be added.

4.1.3 Representation of PS in X3D

In X3D, PS can be represented using next nodes.

- CADLayer / CADAssembly / CADPart / CADFace: parent-child relations
- Transform / ClipPlane : transform and reveal geometric information
- Inline : external referencing to a data file

The methods for PS representation in X3D exactly correspond to those in STEP introduced in 4.1.1.

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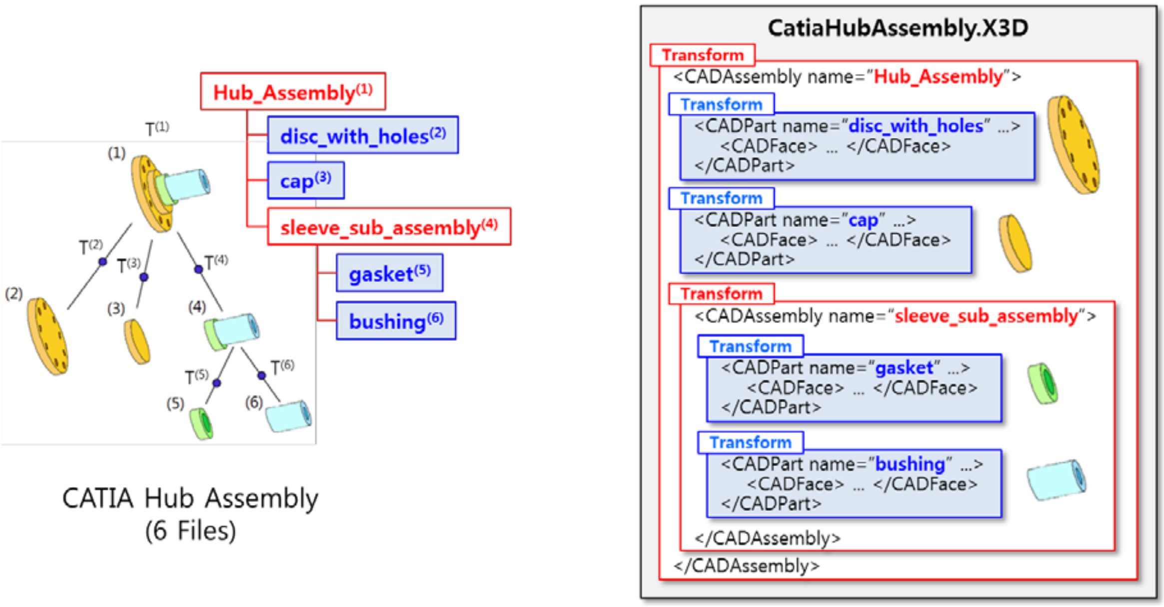
a) assembly and part geometries in the same file

b) external reference

c) nested external reference

4.1.3.1 Assembly and part geometries in the same file



Transform information is usually applied to the leaf nodes which include geometry information. For example, the bushing transform includes all its parents transforms, $T^{bushing} = T^{(1)}*T^{(4)}*T^{(6)}$, and then applies to the geometry data. In this case transform node is not required. PS is represented with CADAssembly and CADPart nodes, and geometry with CADFace node.

Transform information can be conserved separately as is in CAD assembly. In this case, transform node should be used for each assembly or sub-assembly node. For this, a proxy CADAssembly is recommended to be used for a CADPart.

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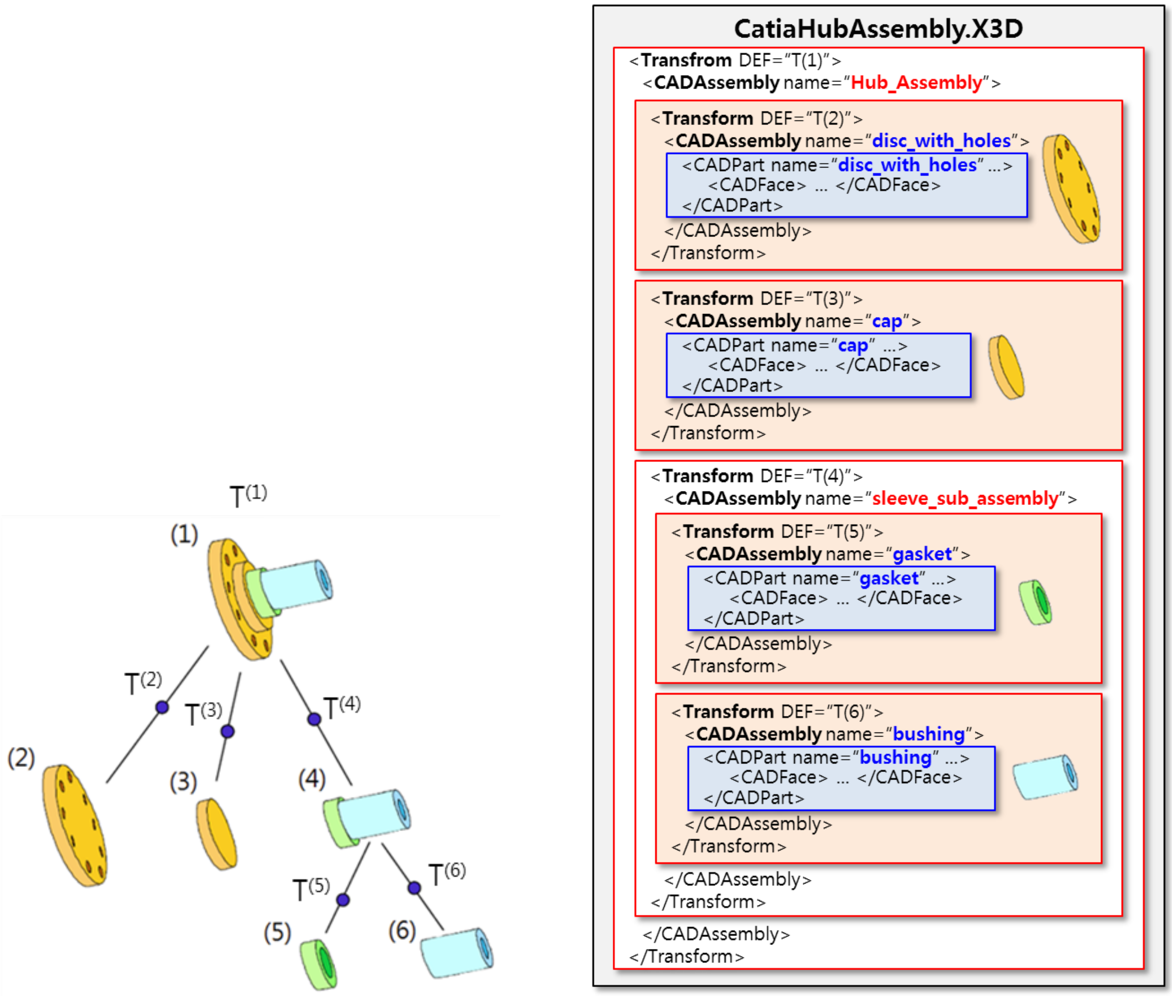

4.1.3.2 External reference

For the external reference in X3D, an assembly file has all the PS information and external references for externally saved geometry files, and the geometry files have shape information only. PS is represented as in 4.2.1.1 and the geometry files are referenced using Inline node.

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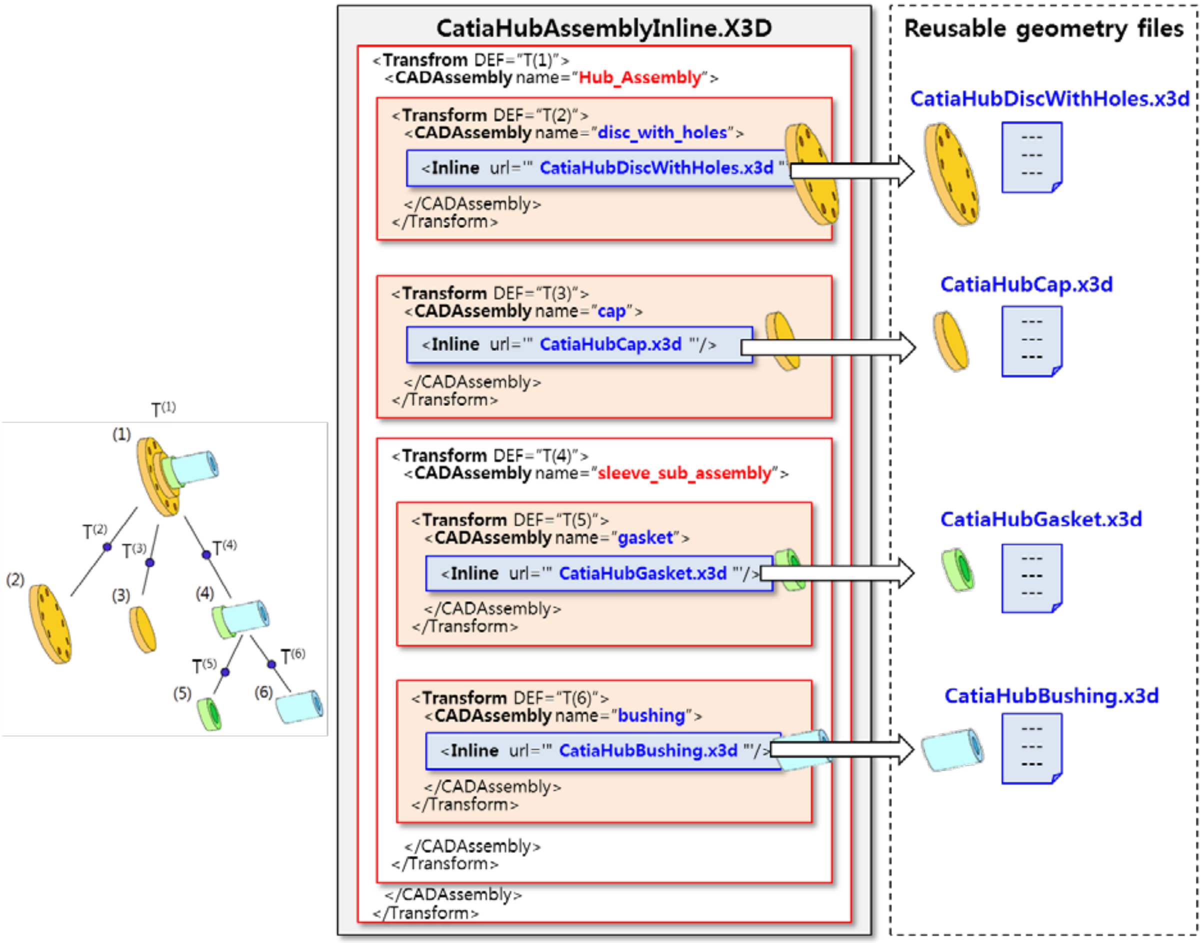

4.1.3.3 Nested external reference

For the nested external reference, each of the root assembly, sub-assemblies, and parts is represented in each file as is in the CATIA Hub Assembly. An assembly or a sub-assembly has PS and external reference for their children only.

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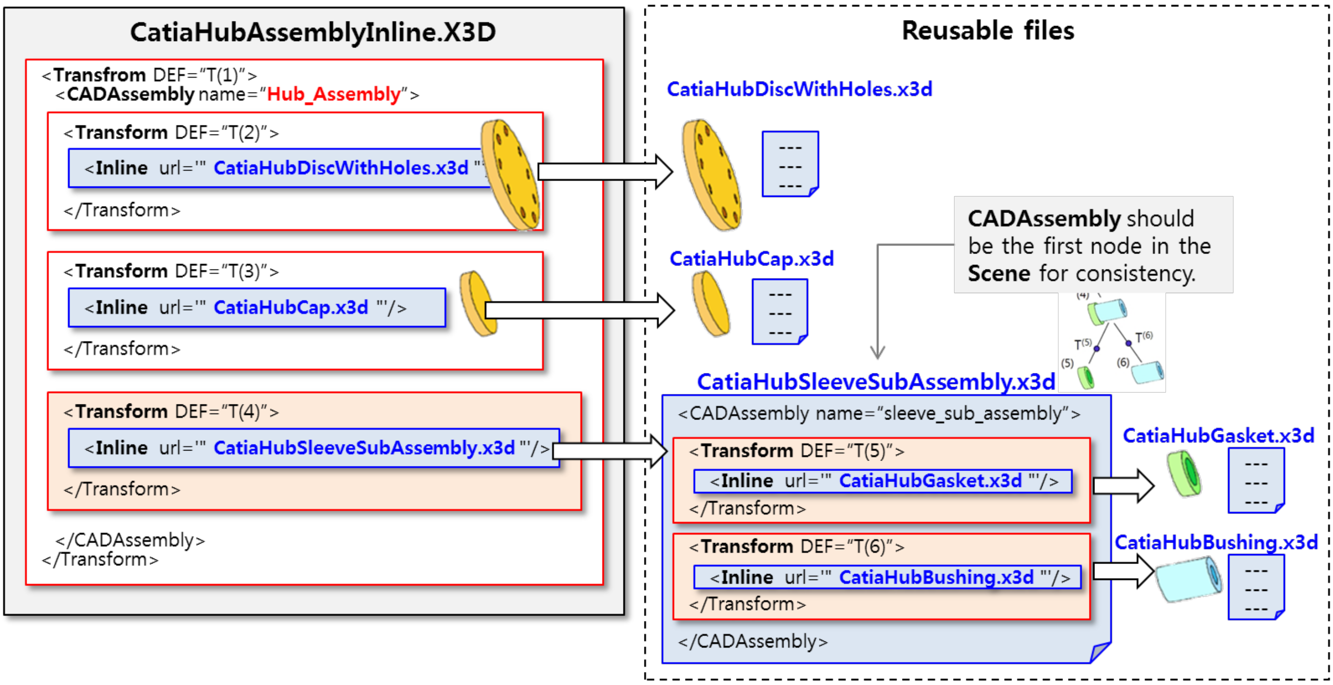

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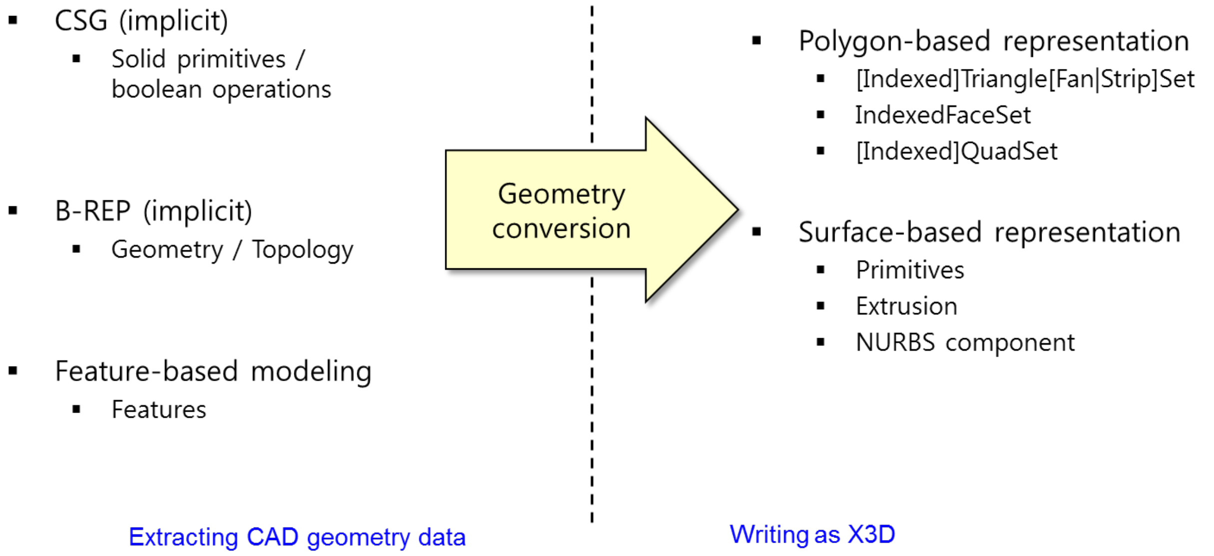
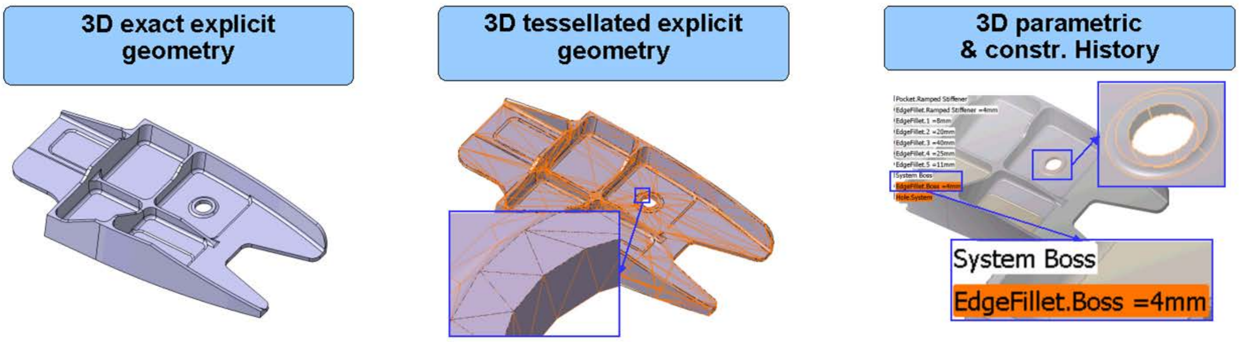
4.2 Geometry



To be specified as text…

4.2.1 Representation of Geometry in STEP



- Exact shape representation (STEP AP 214 and 203)

- Tessellated shape representation

- Parametric Representation

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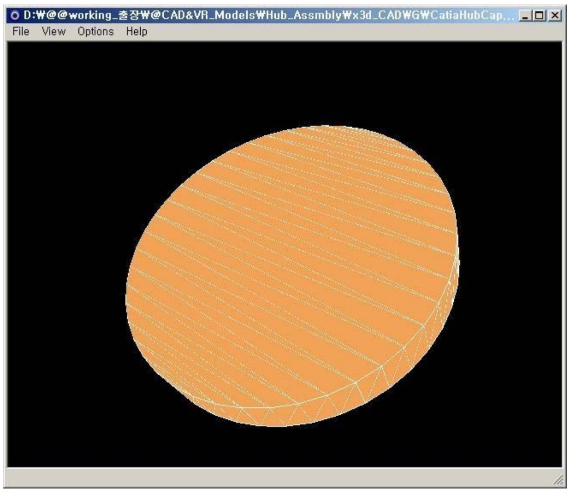
4.2.2 Representation of Geometry in X3D

4.2.2.1 Polygon-based representation

- [Indexed]Triangle[Fan|Strip]Set
- IndexedFaceSet
- [Indexed]QuadSet

‘CatiaHubCap’ Shape with IndexedFaceSet



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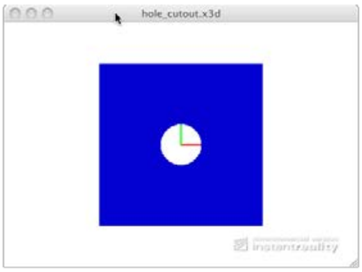
```
<Scene>
  <Viewpoint description='Hello CatiaHubCap' position='0 0 1000'/>
  <CADPart name='cap'>
    <CADFace name='cap_mesh01'>
      <Shape containerField='shape'>
        <Appearance>
          <Material DEF='_material1' diffuseColor='0.94902 0.635294 0.341176'/>
        </Appearance>
        <IndexedFaceSet coordIndex='0 1 2 -1 1 3 2 -1 2 3 4 -1 3 5 4 -1 4 5 6 -1
          <Coordinate point='-75.0 0 0.0 -75.0 0 20.0 -74.2366 10.6736 0.0 -74.23
          <Normal vector='-1.0 0.0 0.0 -1.0 0.0 0.0 -0.989822 0.142308 0.0 -0.989
        </IndexedFaceSet>
      </Shape>
    </CADFace>
    <CADFace name='cap_mesh02'>
      <Shape containerField='shape'>
        <Appearance>
          <Material USE='_material1'/>
        </Appearance>
        <IndexedFaceSet coordIndex='0 1 2 -1 1 3 2 -1 2 3 4 -1 3 5 4 -1 4 5 6 -1
          <Coordinate point='75.0 0.0 0.0 75.0 0.0 20.0 74.2366 -10.6736 0.0 74.2
          <Normal vector='1.0 0.0 0.0 1.0 0.0 0.0 0.989822 -0.142308 0.0 0.989822
        </IndexedFaceSet>
      </Shape>
    </CADFace>
```

4.2.2.2 Surface-based representation

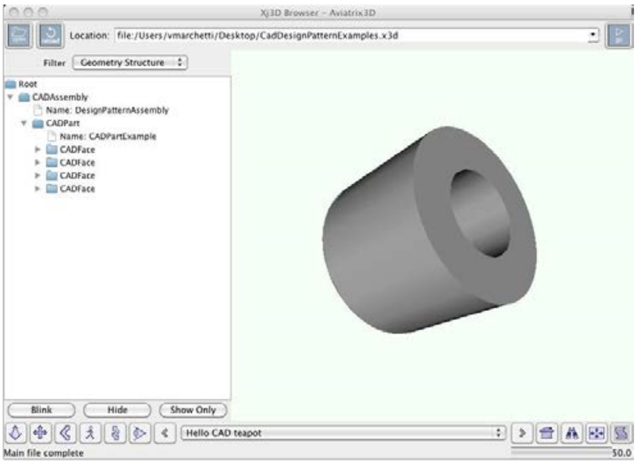
- Primitives

- Extrusion

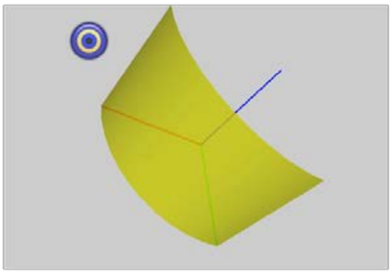
- NURBS component



Rendering of NurbsTrimmedSurface node



Rendering of a cylinder with a hole using Xj3D



Rendering of NurbsPatchSurface

```
<CADAssembly name='DesignPatternAssembly'>
  <CADPart name='CADPartExample' rotation='0 1 0 0' translation='0 0 0'>
    <!-- the rotation and translation attributes serve to position the bushing within the assembly -->
    <CADFace name='outerSurface'>
      <Shape containerField='shape'>
        <!-- note solid='true' and so one-sided external rendering only -->
        <Cylinder bottom='false' height='3' radius='2' top='false'/>
        <Appearance DEF='AppearanceGrey'>
          <Material/>
        </Appearance>
      </Shape>
    </CADFace>
    <CADFace name='innerSurface'>
      <Shape containerField='shape'>
        <!-- note solid='false' and so two-sided internal + external rendering -->
        <Cylinder bottom='false' height='3' radius='1' solid='false' top='false'/>
        <Appearance USE='AppearanceGrey'/>
      </Shape>
    </CADFace>
    <CADFace name='topCap'>
      <Transform containerField='shape' rotation='1 0 0 1.5708' translation='0 1.5 0'>
        <Shape>
          <Disk2D innerRadius='1' outerRadius='2'/>
          <Appearance USE='AppearanceGrey'/>
        </Shape>
      </Transform>
    </CADFace>
    <CADFace name='bottomCap'>
      <Transform containerField='shape' rotation='-1 0 0 1.5708' translation='0 -1.5 0'>
        <Shape>
          <Disk2D innerRadius='1' outerRadius='2'/>
          <Appearance USE='AppearanceGrey'/>
        </Shape>
      </Transform>
    </CADFace>
  </CADPart>
</CADAssembly>
```

4.2.3 Geometry Conversion

Mapping of STEP vocabularies toexisting parametric X3D nodes to be specified here…

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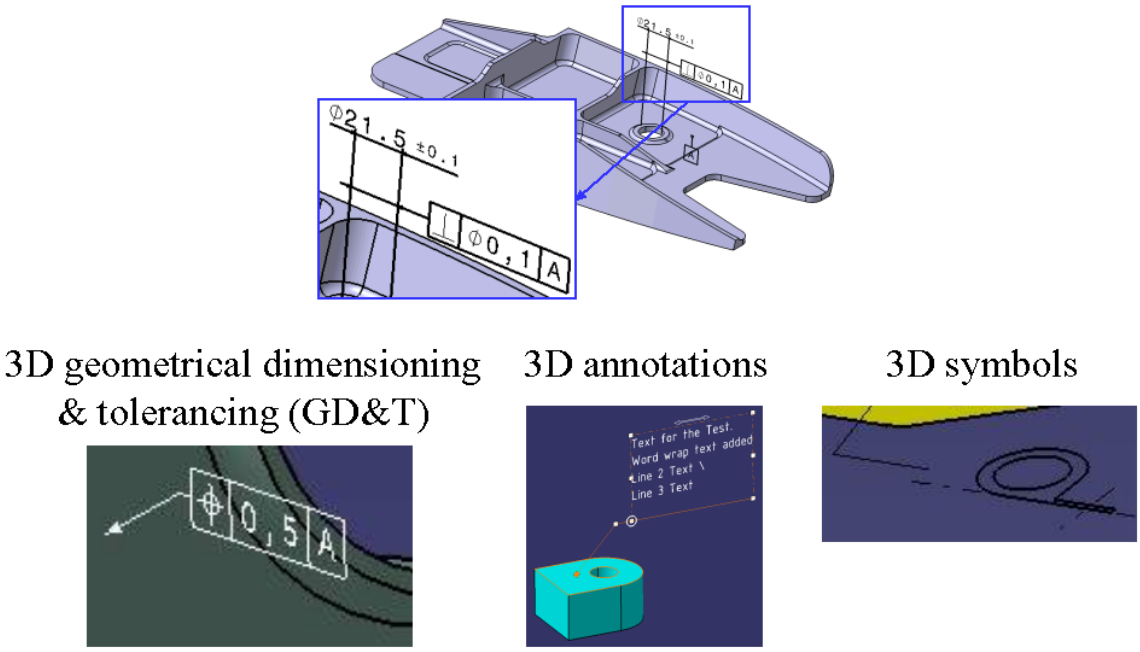
4.3 Product Manufacturing Information (PMI)

4.3.1 PMI representation in STEP

PMI : Geometry Dimension & Tolerance(GD&T) / annotations / symbols



PMI representation in STEP [3]

- Graphic representation

- Semantic representation

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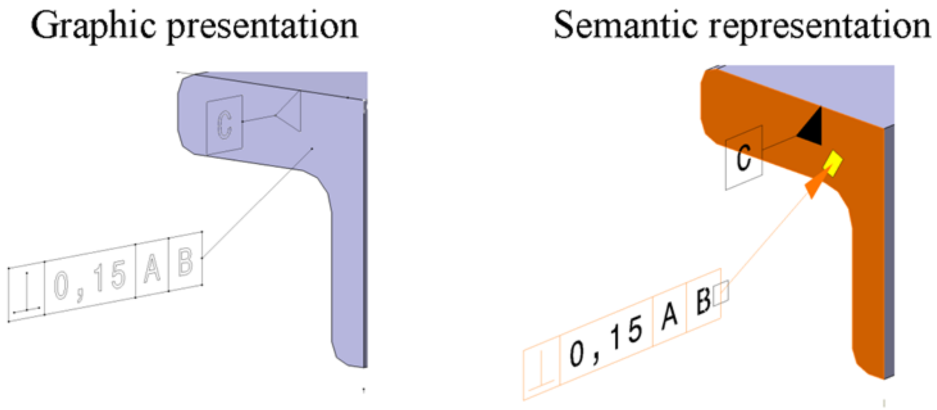

4.3.2 PMI representation in X3D

4.3.3 Conversion of STEP 3D PMI into X3D

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4.4 3D Printing and 3D Scanning

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5. Conclusions and Recommendations

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Bibliography

[1] Recommended Practices for External References with References to the PDM Schema Usage Guide, Release 2.1, CAx Implementation Forum and PDM Implementation Forum, January 19, 2005.

[2] STEP AP242 Project, http://www.ap242.org/geometry-assembly-pmi-interoperability

[3] LOTAR, LOng Term Archiving and Retrieval, http://www.lotar-international.org/

[4] Recommended Practices for Geometric Dimensions & Tolerances (GD&T) “Polyline Presentation”, Release 1.0, CAx-IF & LOTAR, June 16, 2008

[5] Test Suite for the CAx Implementor Forum Round 34J April-September 2014, Release 1.0, CAx-IF, June 30, 2014.

[6] Cho, G., Hwang, J., and Kim, Y., “Translation of 3D CAD Data to X3D Dataset Maintaining the Geometry and Structure Information of a Product.” *The Transactions of the Korea Information Processing Society*, VOL. 18-A, NO. 03, PP. 0081-0092, June 2011.

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